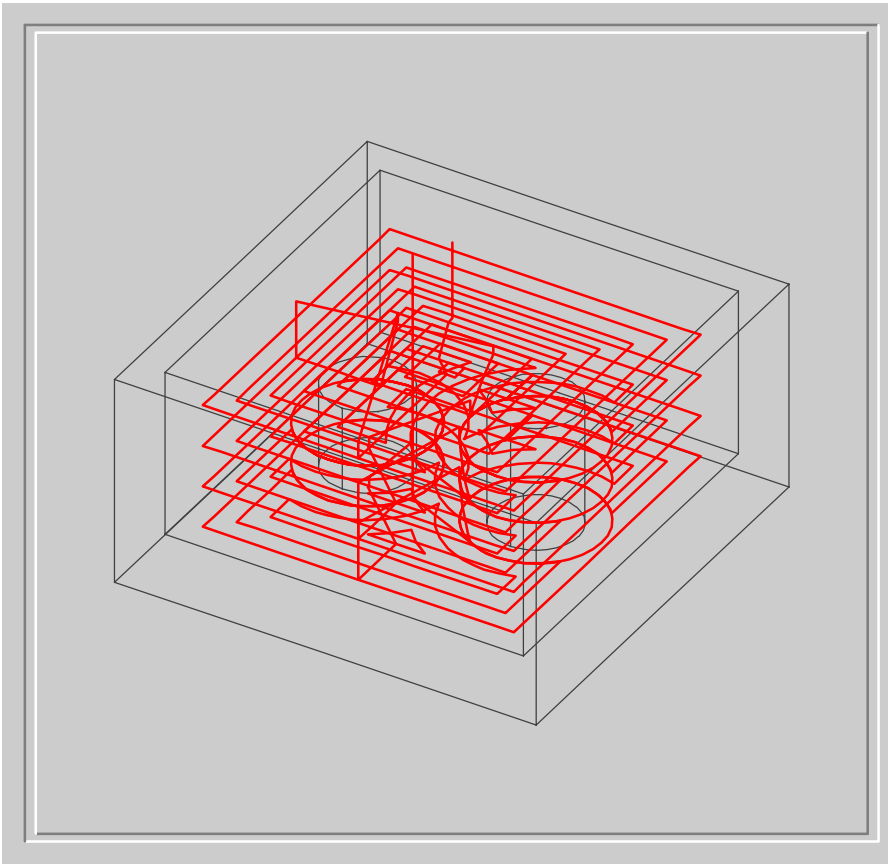


Introduction to Generative Machining

I-DEAS™ Tutorials: Milling Projects and Turning Projects

I-DEAS Generative Machining™ generates NC toolpaths for 2-, 2 1/2-, and 3-axis milling operations, 5-axis positioning, 2-axis turning, and the most commonly used hole-making operations. The software includes tools for planning all the manufacturing steps that result in a finished part.

In this tutorial, you'll learn the basic process for planning a job.

Learn how to:

- create an NC job
- create an operation
- define critical depths
- create a finish operation
- view all the toolpaths

Before you begin...

Prerequisite tutorials:

- all tutorials under the Modeling Fundamentals menu

The file you need for this tutorial is distributed with the product. You must copy it into your local directory.

Move to the local directory where you want to copy the file. Then:

In UNIX:

```
cp $SDRC_INSTL/examples/nc/tut_nc_intro.arc .
```

In Windows use:

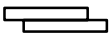
```
copy %SDRC_INSTL%\examples\nc\
tut_nc_intro.arc .
```

If you can't copy the file, you may have to set up the variable needed to copy from the I–DEAS installation.

```
. sdrc_oadev
```

If you can't access the file, contact your system administrator. The file may not be installed.

If you didn't start I-DEAS with a new (empty) model file, open a new one and name it ncstart.


File
Open

Open Model File form


Model File name: ncstart

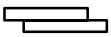

OK

Make sure you're in the following application and task:


Application: Manufacturing


Task: Generative Machining

Set your units to inches.


Options
Units


Inch (pound f)

Import the archive file that contains the parts and tools that you need to complete this tutorial. Importing an archive file can take several minutes. Be patient.

File
Import

Import Selections form


I–DEAS Archive File
OK

File Name Input form

Archive file name: tut_nc_intro.arc
OK

The Manufacturing application quits, an informational message is displayed (the message will dismiss automatically), and the archive file is imported.

Import Archive File Status

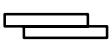

Check *I-DEAS List*.

Be sure to check the List region to be sure that the parts imported properly.

Dismiss

A second informational message is displayed (the message will dismiss automatically) and the Manufacturing application starts.

Recovery Point


File
Save

Warning!
If you're prompted by I-DEAS to save your model file, respond:


No

Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

If you make a mistake at any time between saves and cannot recover, reopen your model file to the last save and start over from that point.

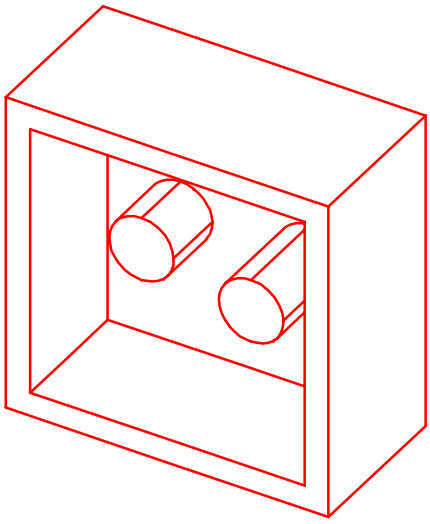
Hint
To reopen your model file to the previous save, press Control-Z.

An NC job represents your entire job plan. It's the collection of setups, opgroups, and operations required to machine a part.

- A setup represents one fixtured configuration of the stock, fixture, and clamps on the machine table.
- An opgroup is a container for grouping related operations.
- An operation contains all the information needed to create the toolpath used to generate a standard CL file.

When you create a job, the software creates the first setup and opgroup. You will add the operations to fill the opgroup and any additional opgroups and setups necessary to complete the job.

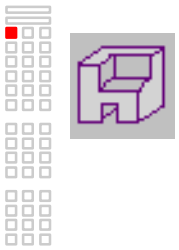
When you delete the job, all the setups, opgroups, operations, and other job entities are deleted too.



In this section you'll create an NC job and use the software's tools to check the contents of your model file. You'll also add a part instance to the setup and use the software's tools to check the setup assembly.

What: Create an NC job and check what is added to your model file.

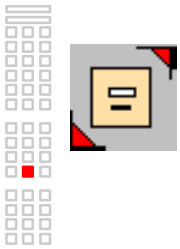
How:



NC Job Create form

Job Name: Block Job

OK



Manage Bins form

View: All

Things to notice

The following entities were added to the model file when the NC job was created:

- Block Job—the NC job.
- Setup-1_Assembly—the NC Setup. Later, this will include the first configuration of part instances representing the machine table, stock, clamps, and other fixtures.
- The other elements in the list were added when the archive file was imported.

Dismiss

What: Add a part instance to the setup.

How:

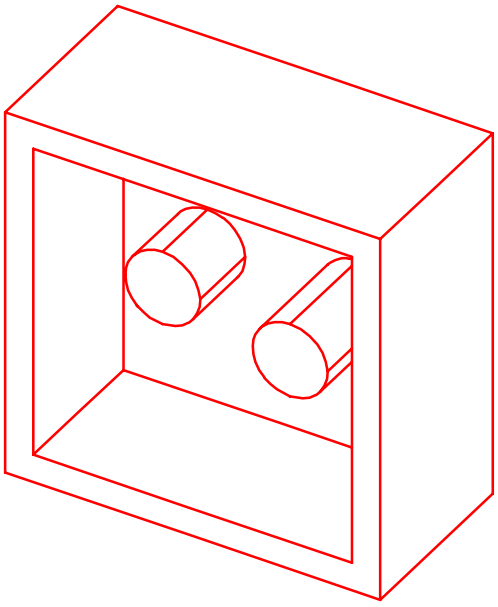
Get

From Bin/Library

Select Part/Assembly form

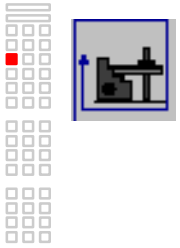
tut_block

OK



What: Check the contents of the setup assembly.

How:



Setup Specification form

Things to notice

The box on the right side of the form lists the part instance you added to the setup.

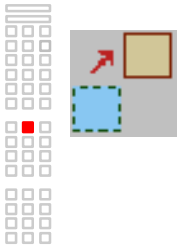
Dismiss

Recovery Point

File
Save

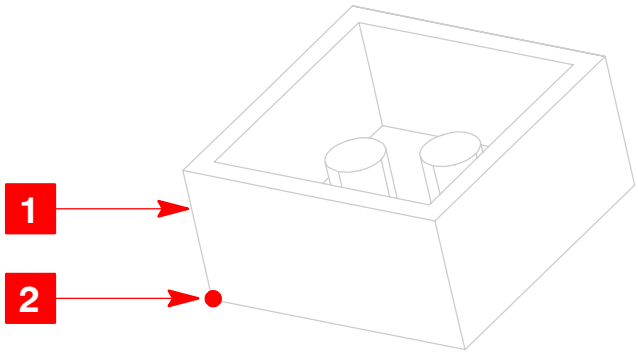
What: Move the part to the global space coordinate system. The global space coordinate system acts as the origin, or program zero, of the job. In later tutorials, you'll learn how to define a machine instance and specify indexing.

How:



1 anywhere on the part

Move To



2 V4

Key In

Check *I-DEAS Prompt*.

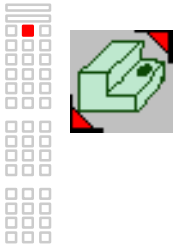
Enter 0, 0, 0.

(accept)

(done)

What: Bring up the NC Job Planning form to see the job you created.

How:



Things to notice

The form contains a Setup–1 and an indented OpGroup–1. This happens automatically when the NC job is created.

For more complex jobs, you'll need to create any additional setups or opgroups on your own.

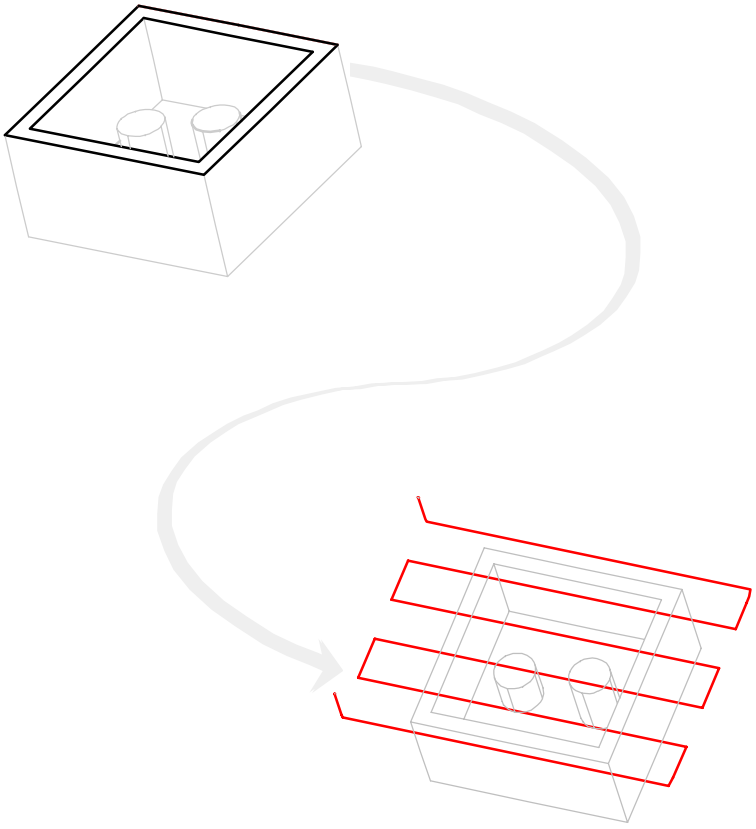
Don't close the NC Job Planning form.

An operation contains the information needed to generate toolpaths. For a basic operation, you can define the following:

- the surfaces to be machined
- the machining coordinate system (covered in later tutorials)
- bounding sections (covered in later tutorials)
- the tool
- machining parameters that specify how to cut the part
- the toolpath itself

The icons for completing these tasks are located on the Operation Specification form.

In the next steps, you'll create a face mill operation to machine the face of the part. When creating the operation, you'll define a bi–directional cut pattern and a cut type of *Climb First*.



What: Modify the opgroup by adding an operation to machine the top of the part.

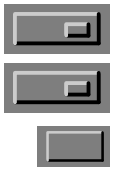
How:

NC Job Planning form

OpGroup–1

OpGroup Specification form

Operation Selection form



Category: *Milling*

Type: *Face Mill*

Create

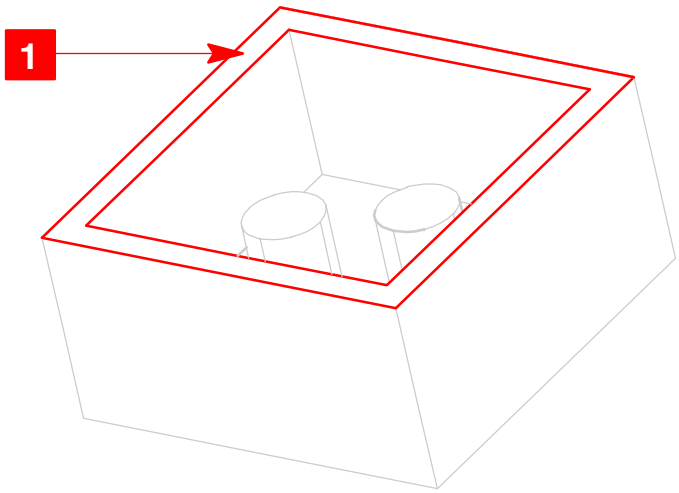
Don’t close the Operation Specification form.

What: Name the facing operation and select the surface to be machined.

How:

Operation Specification form

Name: Face Mill Top of Part



Hint Use F1, F2, or F3 to position the part for easier selection.

Don’t close the Stock Specification form.

What: Define the stock. You'll learn more about techniques for defining stock in later tutorials.

How:

Stock Specification form

 *System Defined Stock*: On

 *With XY Offset of*: 50

 *Stock Top*: 0.25

 *Stock Top*

The stock top is already selected, so just accept.

 *OK*

 Don't close the Operation Specification form.

What: Create a tool for the facing operation.

How:

Operation Specification form

Cutting Tool Specification—Mill form

Identifier: 1.5 inch Face Mill

Press the Tab key to advance to the next field.

I-DEAS Warning

OK

Style: *Face Mill*
Shank Diameter: 1
Holder to Tip Distance: 2.5
Max Depth of Cut: 2
Cutter Diameter: 1.5
Nose Radius: 0

Things to notice

The shape of the tool on the form changes as you define its parameters.

OK

Don’t close the Operation Specification form.

What: Define a bi–directional cut pattern and the cut type as *Climb First*.

How:

Operation Specification

Machining Parameters: Cut form

 *Cut Pattern*: *Bi–Directional*

 *Cut Type*: *Climb First*

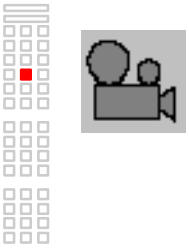
 *OK*

Don’t close the Operation Specification form.

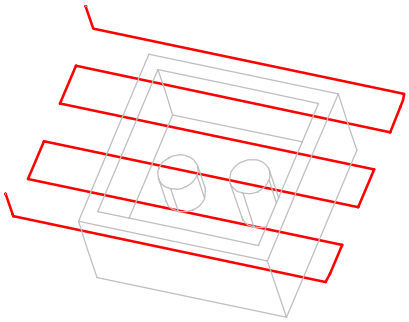
What: Create and animate the toolpath.

How:

Operation Specification form



Animate Tool form

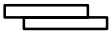


Things to notice

Because of the bi–directional cut pattern and the cut type of *Climb First*, the tool stays in contact with the surface. Also, notice that the tool goes past the edge of the surface by 50 percent of its diameter because *With XY Offset Of* is set to 50%.

 *Dismiss*

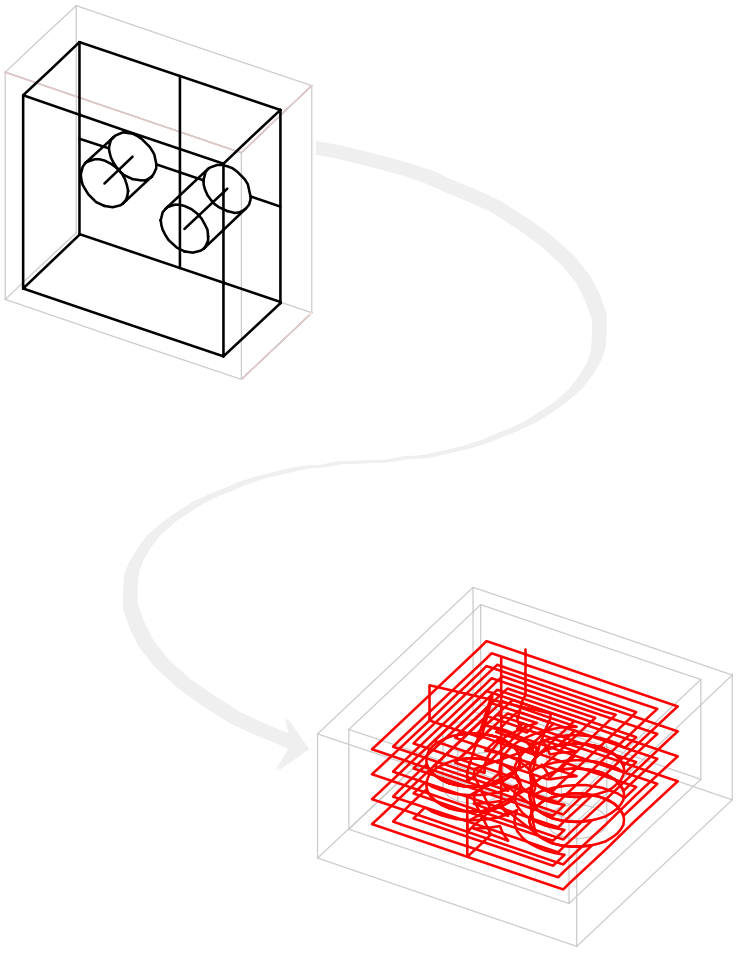
Recovery Point

 *File*
Save

In the next few steps, you'll define critical depths for an operation. When generating a toolpath, the software creates a series of axial depths of cut. An axial depth that contacts a selected planar surface, whose angle to the tool is less than the critical depth angle, is a critical depth.

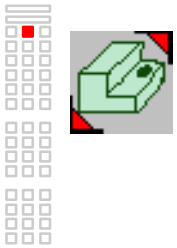
You can control the number of critical depths by which surfaces you select. First, you'll select the surfaces composing the cavity—except for the tops of the bosses—and generate a toolpath. Thus, you'll have only one critical depth for the bottom surface of the cavity.

Then you'll modify the operation and pick the tops of the bosses. These surfaces will add two critical depths to your toolpath.



What: Create an opgroup.

How:



NC Job Planning form

Deselect *Face Mill Top of Part* by pressing the Control key and selecting *Face Mill Top of Part*.

Things to notice Notice on the OpGroup Specification form, OpGroup–2 is created. Because you deselected the operation and opgroup on the NC Job Planning form, the new OpGroup–2 is placed at the end of the Setup–1.

Don’t close the OpGroup Specification form.

What: Create a volume clear operation. This operation will remove most of the material from the cavity.

How:

OpGroup Specification form

Operation Selection form


Category: *Milling*


Type: *Volume Clear*


Create


Don’t close the Operation Specification form.

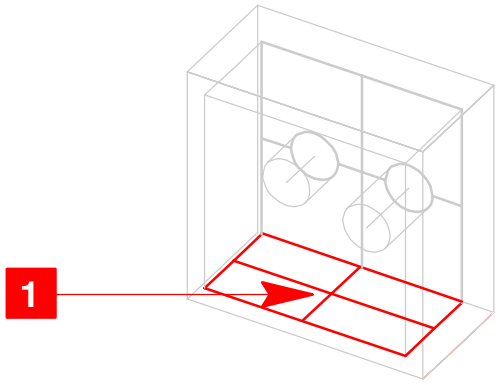
What: Name the volume clear operation and select the surface to be machined.

How:

Operation Specification form

Name: Cavity Milling

1 double–click on the face to select the cavity



Things to notice Although all the surfaces composing the cavity appear to be selected, the sides and tops of the bosses aren't. The software will avoid the boss surfaces and won't recognize the tops as critical surfaces.

Don't close the Stock Specification form.

What: Define the stock.

How:

Stock Specification form

- *Stock Top*: 2.5
- *Stock Bottom*: .5
- *OK*

Don’t close the Operation Specification form.

What: Create a tool for the volume clear operation.

How:

Operation Specification form

Cutting Tool Specification form

Identifier: .5 inch Flat End Mill

Press the Tab key to advance to the next field.

I-DEAS Warning

OK

Style: *End Mill*
Shank Diameter: .75
Holder to Tip Distance: 4
Max Depth of Cut: 3.5
Cutter Diameter: .5
Nose Radius: 0
OK

Don’t close the Operation Specification form.

What: Set the machining parameters to machine the cavity. Typically, you use a *Spiral Out* cut pattern to mill a cavity so the slotting cut is in the middle of the cavity and the last pass is along the walls.

You'll use an axial entry only because the selected surfaces compose a closed volume. You define an *Along Path* so that the tool enters by following the cutting portion of the toolpath while sinking to the cut depth.

How:

Operation Specification form

Machining Parameters: Cut form

 *Cut Pattern*

 *Cut Type*: *Climb*

 *Axial Depths...*

Axial Depths form

 *Maximum Depth of Cut*: 0.5

 (next to *Maximum Depth of Cut*)

 *OK*

 Don't close the Machining Parameters form.

Machining Parameters: Cut form

Cut...
Axial Entry...

Machining Parameters: Entry form

Entry Type: Along Path
Sink Angle: 60
Axial Entry...
Planar Entry...

Machining Parameters: Entry form

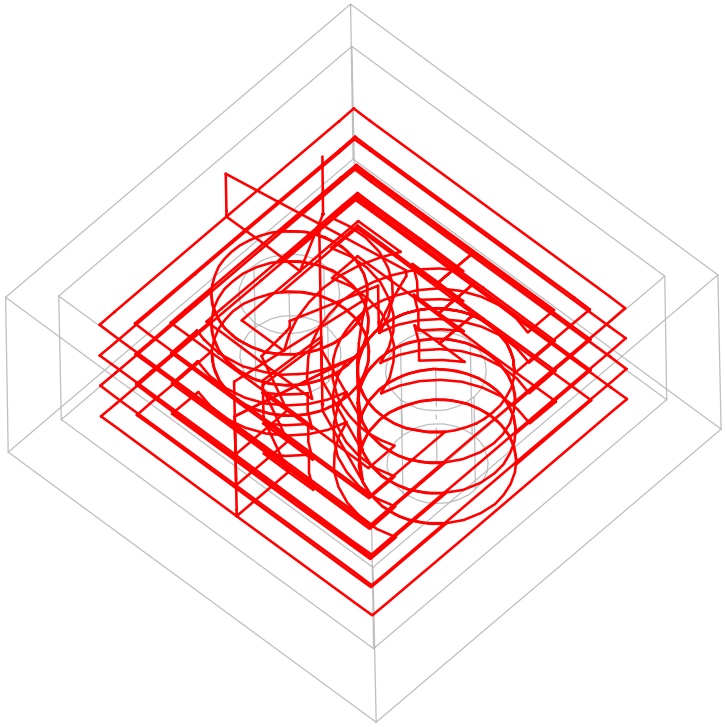
Entry Type: Do Not Use
OK

Don’t close the Operation Specification form.

What: Create the toolpath.

How:

Operation Specification form

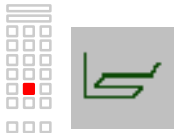


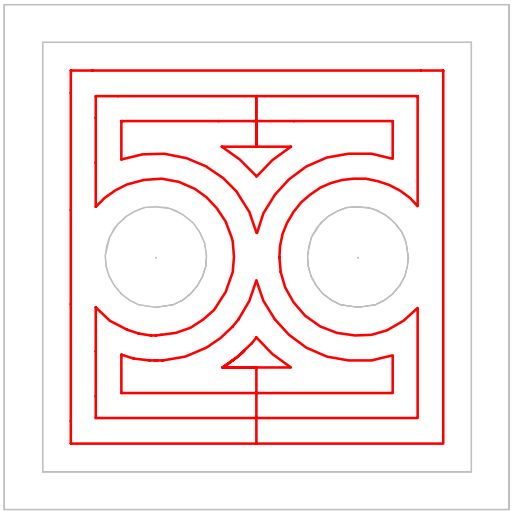
Recovery Point

File
Save

What: View the second depth of the toolpath. Then return to the full view of the toolpath.

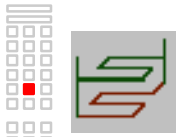
How:





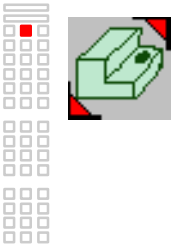
Things to notice

Because you didn't select the tops of the bosses, the software didn't recognize them as critical depths. The tool machined around them. Tool transition moves between each machining region are displayed as separate toolpath regions.



What: Modify the operation to machine the tops of the bosses.

How:



NC Job Planning form

Don’t close the Operation Specification form.

What: Pick the tops of the bosses to be machined.

How: When selecting the tops of the bosses, don't deselect the rest of the cavity.

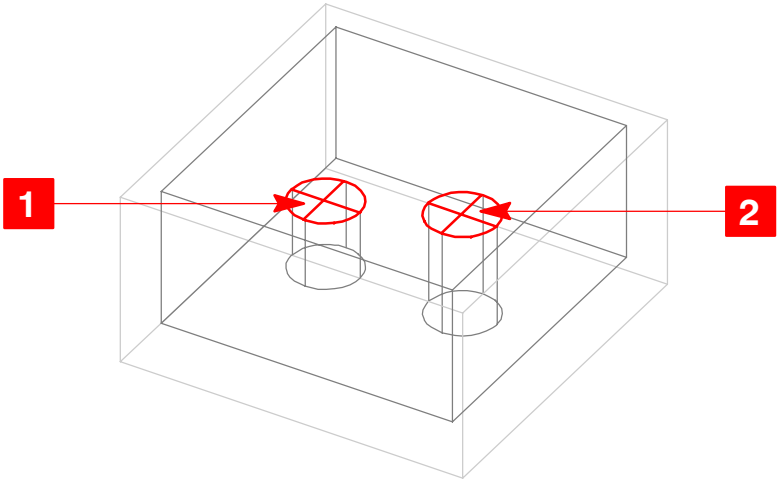
Operation Specification form

Stock Specification form

 *Hide Toolpath*

1 shift double–click on surface to select boss

2 shift double–click on surface to select boss



 *Show Toolpath*

 *OK*

Don't close the Operation Specification form.

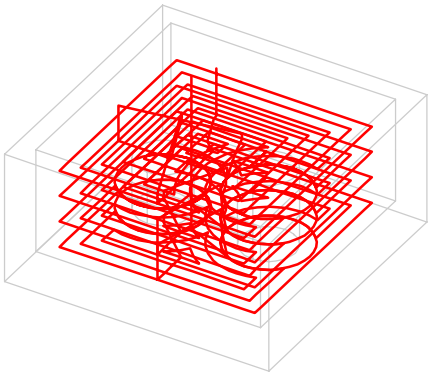
What: Create the toolpath.

How:

Operation Specification form

I-DEAS Warning

OK



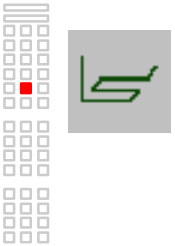
If your toolpath appears drastically different, modify the operation, pick the surfaces again, and regenerate the toolpath.

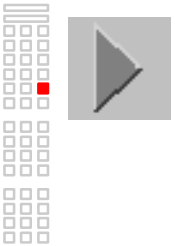
Recovery Point

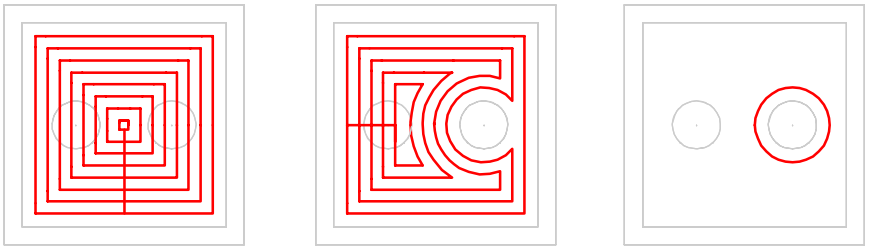
File
Save

What: View the first and second depths of the toolpath. Then return to the full view of the toolpath.

How:

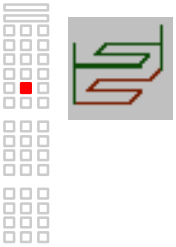






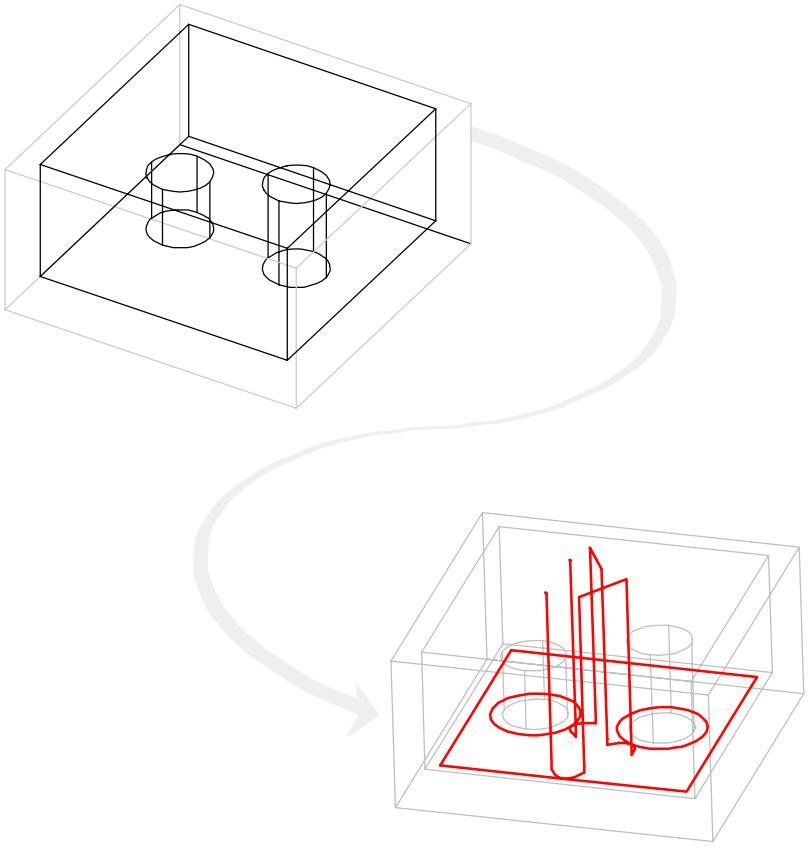
Things to notice

Notice that the toolpath now contacts the tops of the bosses. You now have three critical depths—two on the tops of the bosses and one at the bottom of the cavity. Notice as you cycle through the depths, the software makes a pass around the bosses after roughing.



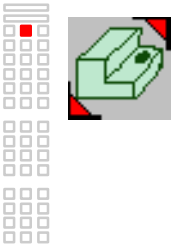
In the previous operation, you removed all the stock from the floors of the part, but left .03 inches of material along the walls. Now you'll create a profile operation to clean up the walls. (You can check the amount of material left by the volume clear operation by modifying it and checking the Machining Parameters—Allowances and Tolerances form.)

You'll also learn how to generate axial depths. Then, you'll delete two critical depths so the tool creates a finish pass along the bottom surface of the cavity.



What: Create a profile operation to finish the cavity and bosses.

How:



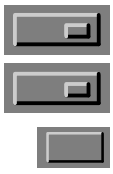
NC Job Planning form

OpGroup–2

OpGroup Specification form

Deselect *Cavity Milling* by pressing the Control key and selecting *Cavity Milling*.

Operation Selection form



Category: *Milling*

Type: *Profile*

Create

Don’t close the Operation Specification form.

What: Name the profile operation and verify that the surfaces are still selected from the previous operation.

How:

Operation Specification form

Name: Profile Walls and Bosses

Surface Selection form

 *Dismiss*

Don’t close the Operation Specification form.

What: Create a tool for the profile operation.

How:

Operation Specification form

Cutting Tool Specification—Mill form

Identifier: .375 inch Flat End Mill

Press the Tab key to advance to the next field.

I-DEAS Warning

OK

Style: *End Mill*

Cutter Diameter: 0.375

Nose Radius: 0

OK

Don’t close the Operation Specification form.

What: Set the depth of cut manually rather than allowing the software to calculate it.

How:

Operation Specification form

Machining Parameters: Cut form

Axial Depths...

Axial Depths form

Maximum Depth of Cut: 2.5


(next to *Maximum Depth of Cut*)

Generate Depths

Things to notice

Notice that three critical depths are generated. The depths still represent the tops of the two bosses and the bottom of the cavity.

Don’t close the Axial Depths form.

What: Delete the first two critical depths. When you generate the toolpath, the software will create a finish pass on the bottom surface only.

Then define the start point for the entry. Note that 2.5 inches is half the length of the part.

How:

Axial Depths form

 2.000000 critical depth

 1.500000 critical depth

 *OK*

Machining Parameters form

 *Cut...*

Entry...

 *Start Near:* 2.5,0,0

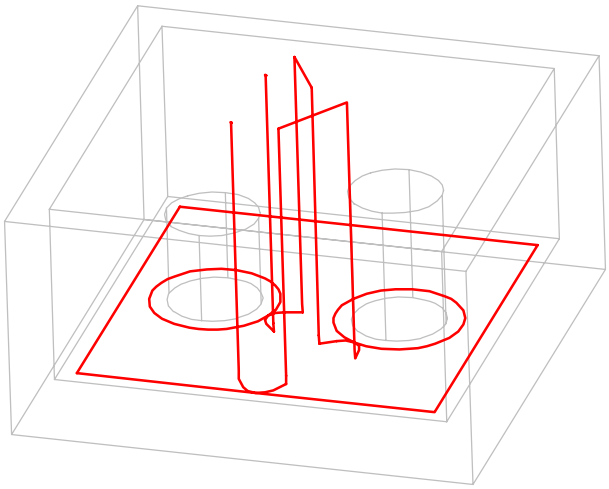
 *OK*

 Don’t close the Operation Specification form.

What: Create the toolpath. Animate the toolpath, if desired.

How:

Operation Specification form



Things to notice

The toolpath has only one depth of cut along the bottom surface of the part.

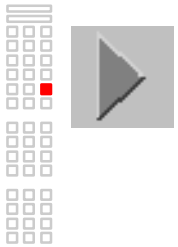
If you animate the toolpath, notice the small axes box located in the upper left corner of the screen displays the exact position of the tool tip as it's shown on the screen. You can use this display to check the actual tool position at points between CL data points.

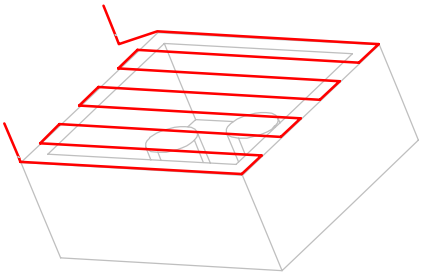
Recovery Point

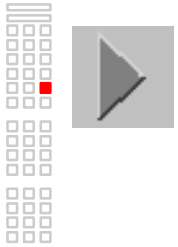
File
Save

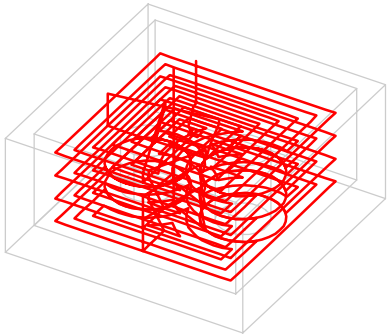
What: Scroll through the toolpaths. You can use these commands to view the toolpaths in your setup quickly or to find an operation to modify.

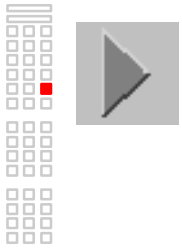
How:

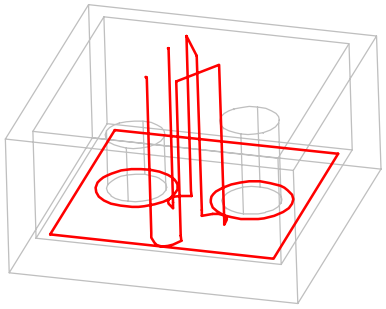












Recovery Point

File
Save

Tutorial wrap-up

You've completed the Introduction to Generative Machining tutorial.

Warning!
Don't delete this model file once you're finished. You'll use this model file and job in a later tutorial.